

Whitepaper

How to Create a Good Quality FE Model

by Cornelia Thieme

How to create a good quality FEM model?

The finite element method (FEM) is increasingly used for structural calculations. For the most part, FEM is used to calculate structural deformation, stress, fatigue life, vibration, or temperature. The manufacturing of a product may be simulated, and one can calculate the acoustics, crash behaviour, and many other disciplines. The FEM model answers questions like "will the product satisfy the design requirements?", "will the component fail?" or later "why did it fail?".

FEM is suitable for structures that go beyond a simple analytical formula representable geometry. The geometry is meshed, that is, divided into many small parts. These parts are called "Finite elements". The FEM solver processes these elements typically using matrix methods to produce the required results (deformation, stress, etc).

Not only FEM professionals, but also designers and developers can quickly deliver results with FEM. But some background knowledge is necessary, in order to produce reliable results. The old adage "garbage in garbage out" applies to FE models!

A linear static FEA calculation requires the following as a minimum set of inputs:

- The meshed geometry
- Material values (Young's Modulus and Poisson's ratio)
- Loads, such as pressure or force
- Restraints. (Usually needed, though methods exist to solve 'unrestrained' models)

A linear static FEA calculation provides as results:

- Deformation
- Stresses. For metal parts, often comparative stresses such as Von Mises may be derived.
- Constraint forces
- Strains, internal forces, and further results derived from the above results

There are variations of this system optimization, E.g.:

- You can as a result determine the maximum force that the design can withstand
- By defining a geometric design space, FE may be used to calculate an optimum component shape

Often, FE modelling begins with the import of CAD geometry into a FEM preprocessor, and with the meshing of the geometry. Different element types can be used.

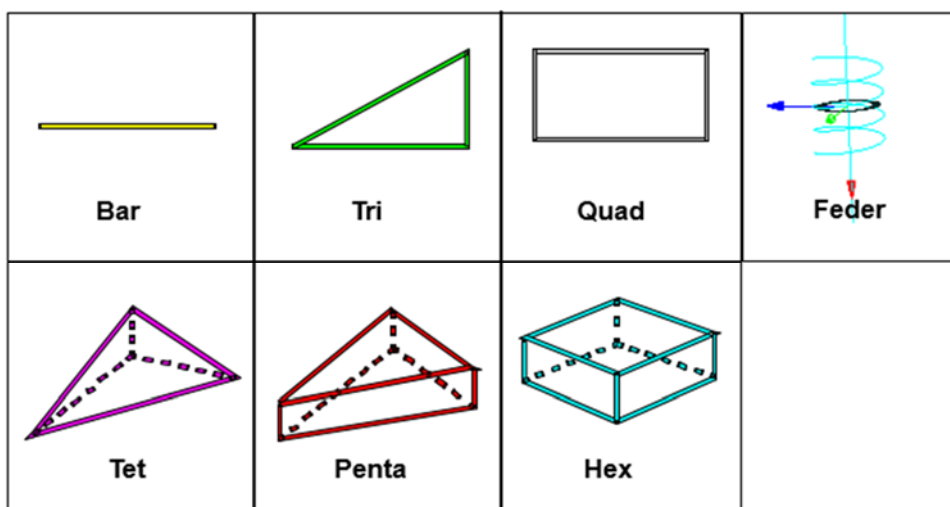


Figure 1: The most commonly used element types

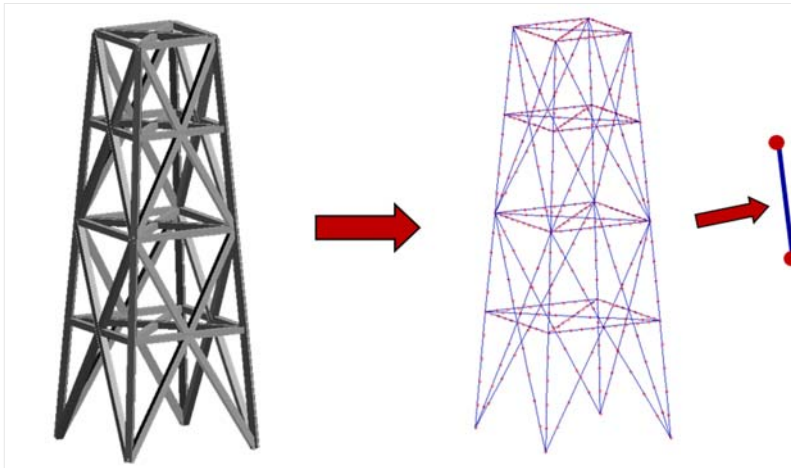


Figure 2: Use of 1D-Elements

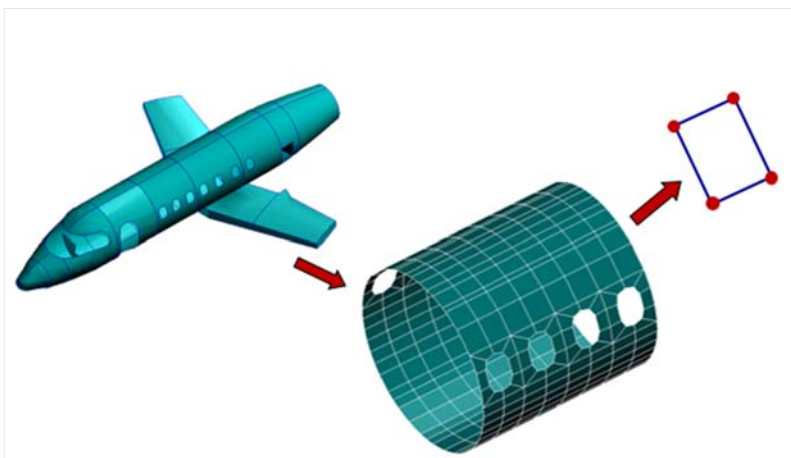


Figure 3: Use of 2D elements

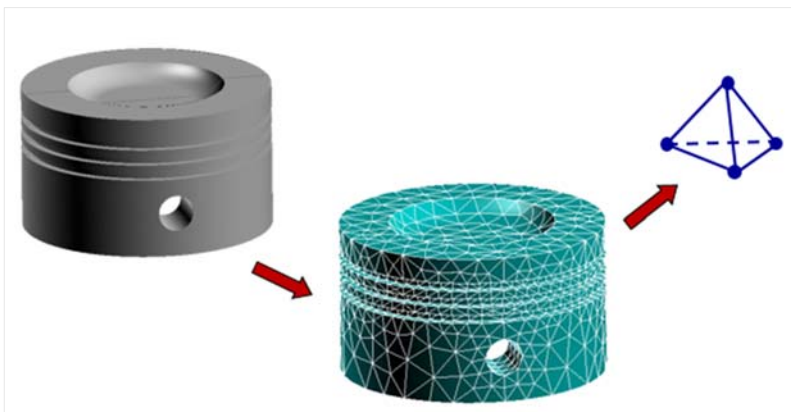


Figure 4: Use of 3D elements

Beam elements are suitable for high aspect ratio structures where one dimension is significantly larger than the other two. Examples include cranes, warehouse structures, or stiffeners (frames, Stringers) in aircraft and ship hulls. Shell elements are suitable for thin wall structures. These structures occur very frequently: aircraft and ship hulls, car bodies, welded constructions in mechanical engineering. For thick walled and solid components, three-dimensional elements are used. E.g. connecting rods, cast/molded brackets, detailed models of bolted joints.

Usually 3D geometry from CAD is available, which can be automatically meshed with tetrahedral elements with a couple of mouse clicks. Why not choose this method for all FE modelling tasks?

- Because of the computing time and accuracy criteria.

If you apply a very fine tetrahedral element mesh to a structure, you will certainly get reliable results (apart from stress singularities, see later). However, the calculation time increases rapidly with the number of elements in the model. It is roughly proportional to the square of the number of degrees of freedom. Degrees of freedom are the possible directions of movement of the nodes. Typically, each node has 6 degrees of freedom: 3 translations and 3 rotations. Depending on the elements that are connected to the nodes, stiffness is only applied to certain degrees of freedom. Degrees of freedom without stiffness are usually eliminated early on by the solver, so that they contribute little to the solve time. It leaves the following degrees of freedom:

- Solid elements (Penta, tetrahedron, hexahedron): 3 degrees of freedom (3 translations)
- Shell elements (triangle, square): 6 degrees of freedom. Sometimes, there is no stiffness applied to the rotation of the shell normal depending on the software and settings, so that the number of degrees of freedom is reduced to 5
- Beam elements: 6 degrees of freedom. There are also bar elements that have only 2 degrees of freedom (translation in longitudinal and torsional)

For example, a tetra or hexa mesh that has 10000 nodes, has 30000 degrees of freedom.

If one halves the element edge length in a hexahedron model, the number of elements increases by eight.

There are also quadratic elements, which have additional nodes at the midpoint of each edge, and use a quadratic calculation approach. The use of quadratic (or second order) elements tends to depend on the solver being used. For tetrahedron models however, quadratic elements are normally recommended for structural calculations, as linear tetrahedron are overstiff.

For thin-walled geometries, in order to calculate the bending stresses accurately, a minimum of 3 solid elements through the thickness should be used. The example below shows a plate clamped at the ends with a pressure load. The resulting deformations are exaggerated. In the middle of the plate (white arrow), the bending stresses are evaluated.

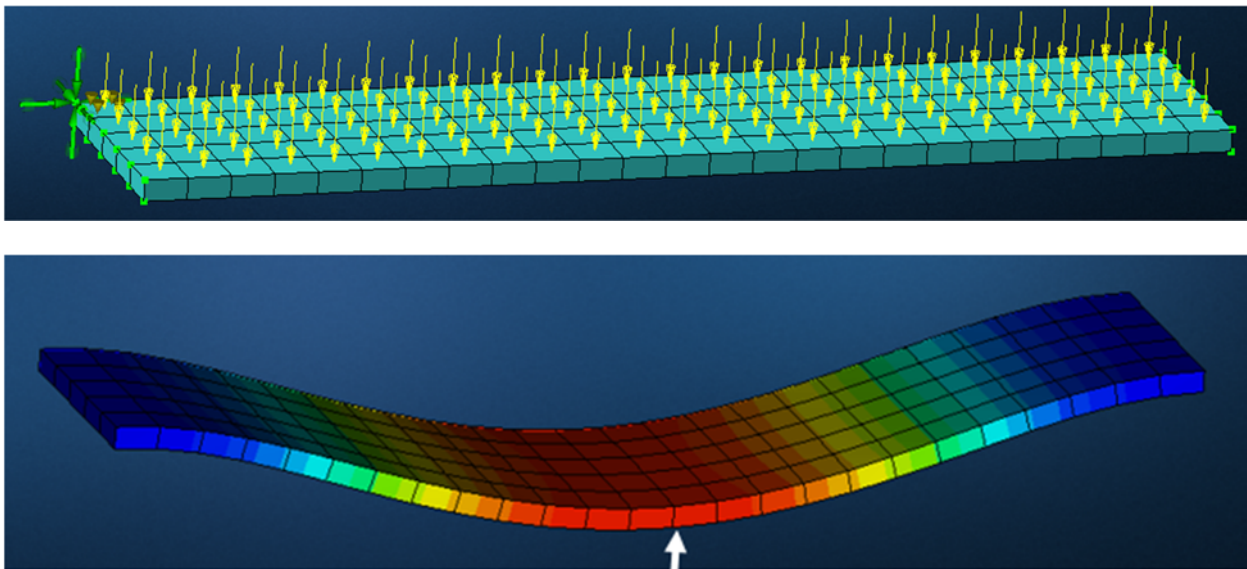


Figure 5: Example for thin-walled geometry - model and results

The graph below shows the variation of bending stress with the number of hexahedron elements through the wall thickness. These are compared to a shell element mesh.

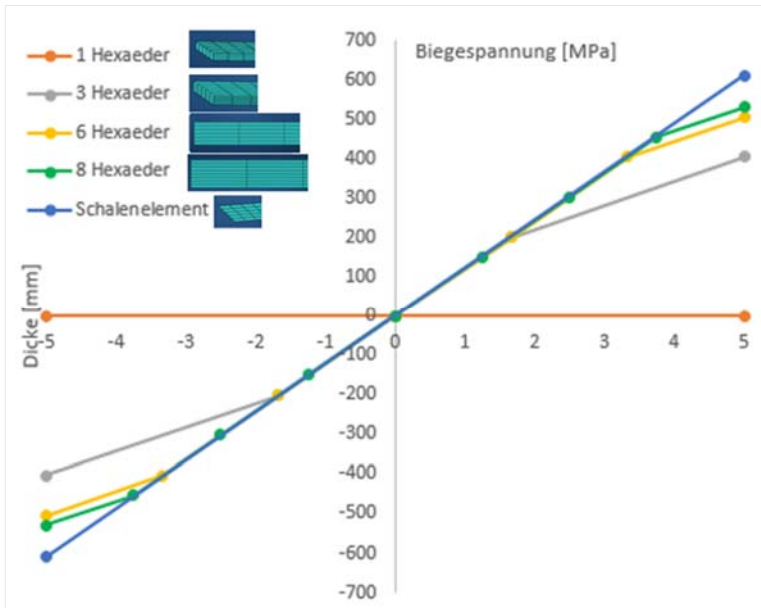


Figure 6: Bending stress on the thickness of the thin-walled geometry of Figure 5

Note: the example uses the stress calculated in the element Center. Using the stress extrapolated on the key points, the results are already with a hexahedral through the wall thickness in the correct range. The extrapolated voltage is not always used, E.g. when non-linear calculation the plastic deformation based on the voltage at the center of the element will be evaluated.

It can be seen that shell elements in this type of model deliver more accurate results, when compared to the lower numbers of hexa elements through the thickness.

Figure 7 shows a thin-walled structure. If you simply meshed it with tetrahedra, you get over 2 million elements – and as can be seen in the yellow plot, there are still walls with less than 3 elements through the thickness. Using shell elements results in 13000 elements, and the calculation time is a few seconds.

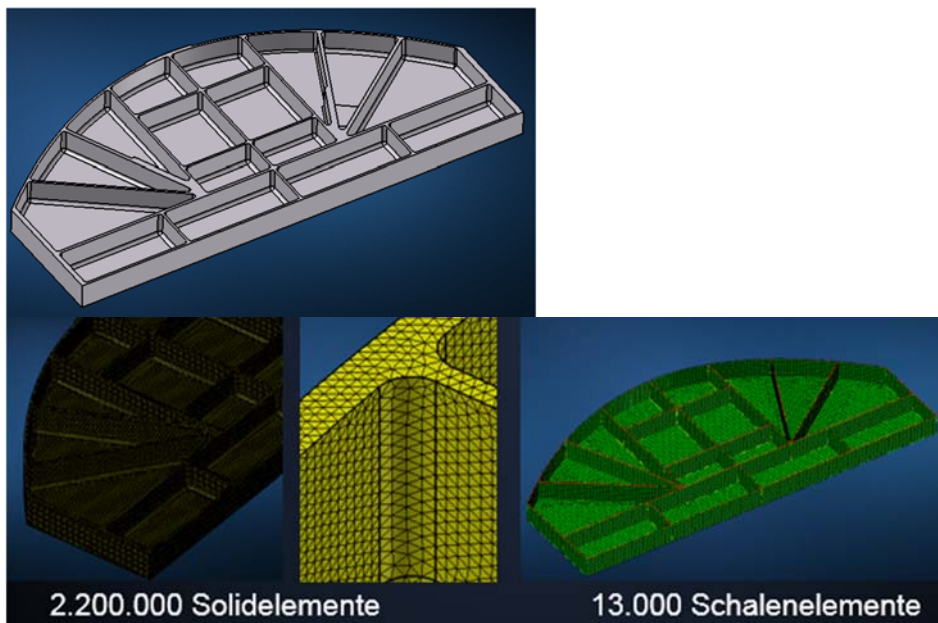


Figure 7: Example of a bulkhead. Above the geometry, below is the mesh with solid and shell elements compared

Therefore, shell elements are typically recommended for thin-walled structures. But how do you make a shell model from the 3D CAD model? The shell elements must lie in the middle of each wall, to avoid artificial inertia effects. Therefore the creation of midsurfaces is an important step in the FEM process.

In MSC Apex, there are efficient ways to extract midsurfaces. In this example, the incremental midsurfacing method is used. The user selects the solid, and Apex generates face pairs between which the midsurfaces are to

be created. The user can then edit them, or simply validate as shown here. Then the midsurfaces are generated, extended, and trimmed to one another automatically.

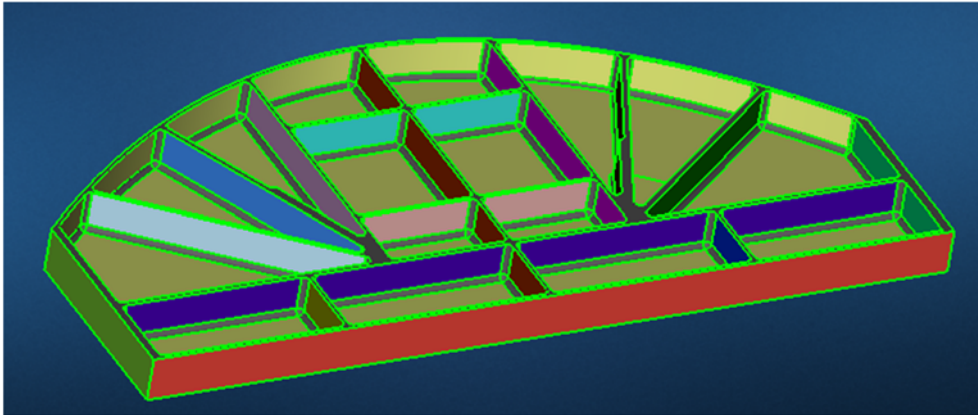


Figure 8: MSC Apex identifies surface pairs for the creation of midsurfaces

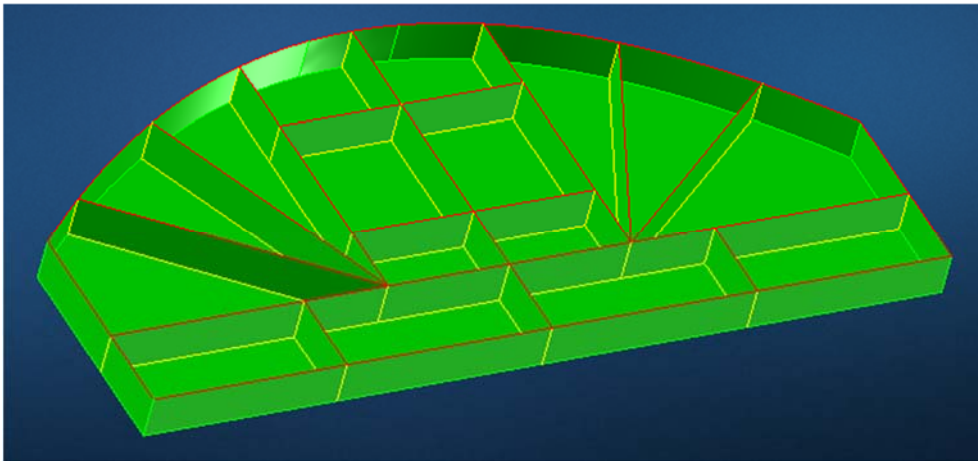


Figure 9: Resulting Mid-surfaces extended and

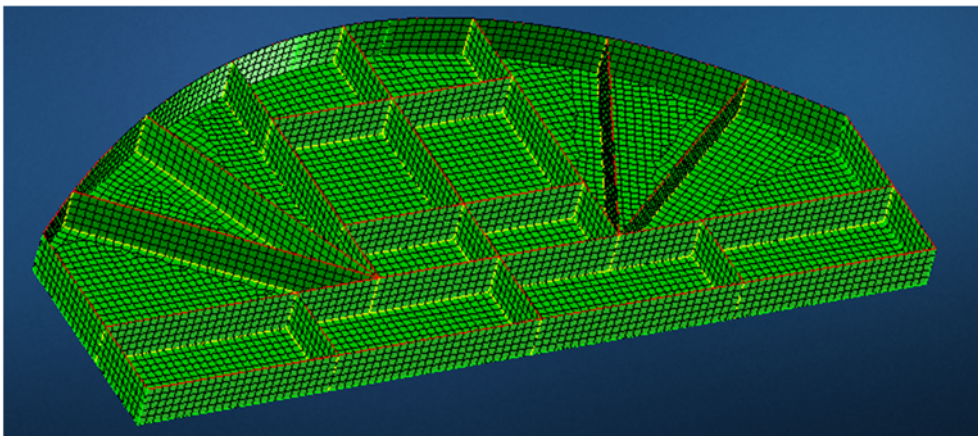


Figure 10: Meshed with shell elements medium surfaces

The wall thickness now needs to be assigned to these shell elements. It is good practice to split the shell where the wall thickness changes, so that there is a correct boundary between different wall thicknesses. In Figure 11, the pink part surface E.g. on the black line is split. The mesh is updated automatically.

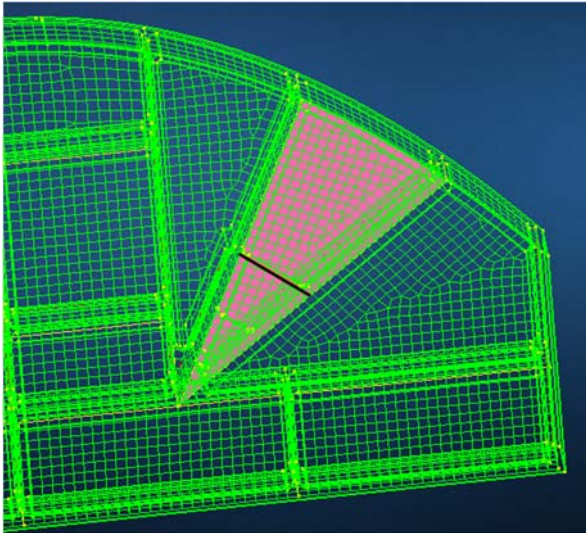


Figure 11: split area

MSC Apex automatically detects the wall thickness.

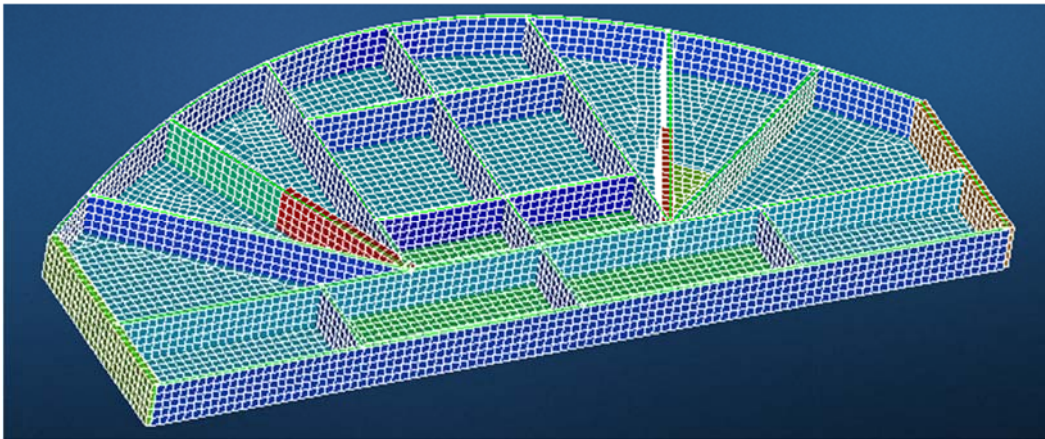


Figure 12: 3D view of the wall thickness

Modeling bar-shaped structures with beam elements makes good sense, as significantly fewer elements are required and the bar forces and moments can be output directly.

If you have a very 3D geometry meshed with tetrahedra, it is worth reducing the number of elements (thereby increasing the solve speed) and improving the quality of the elements by cleaning up the geometry that is to be meshed. MSC Apex can detect and remove short edges, small faces, sliver surfaces, and other geometry problems. Figure 14 shows the mesh before and after cleaning up the geometry. Removal of a short edge leads to a cleaner mesh, avoiding the sliver elements.

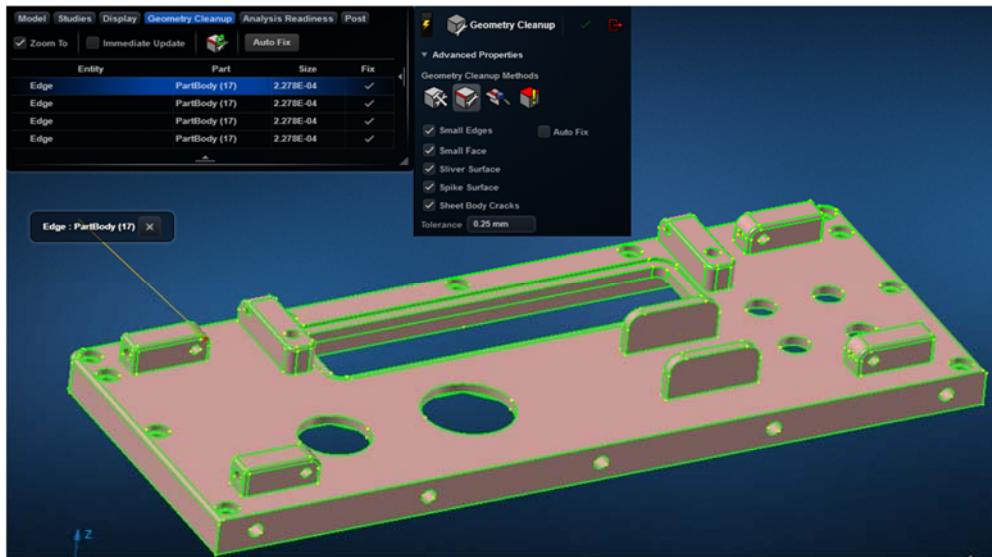
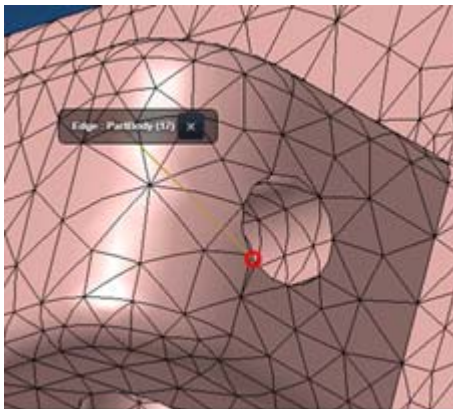
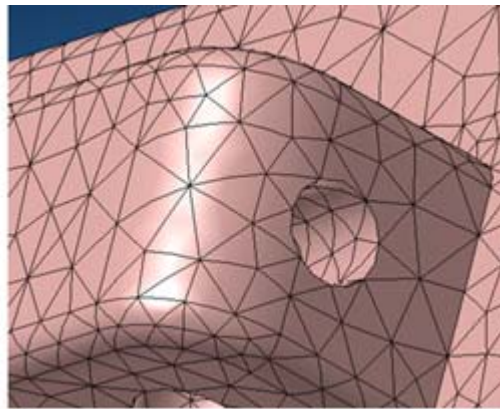


Figure 13: Identification of geometry problems



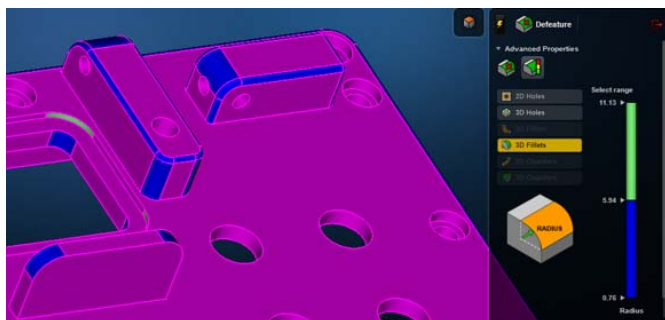
Mesh before clean up



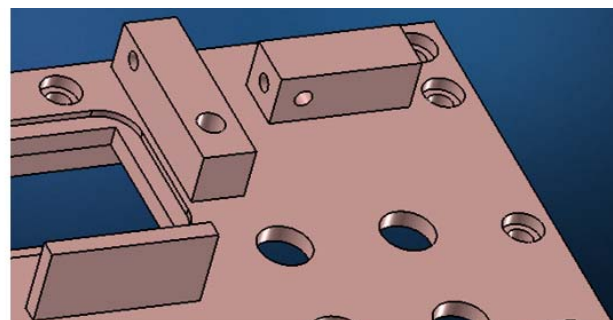
Mesh after clean up

Figure 14: Influence of geometry clean-up on the mesh

In addition to the removal of very small features, it is also useful to remove geometric features that are not important for the calculation. For example the removal of fillets, chamfers or small holes leads to a simpler mesh with fewer elements. Figure 15 shows the automatic identification of fillets with MSC apex. All, or selected fillets can be removed.



Fillets identifizieren



Fillets entfernt

Figure 15: Identify and remove fillets

On the other hand, the mesh can be locally refined in areas where the calculated stresses are of interest. E.g. refined mesh around a hole.

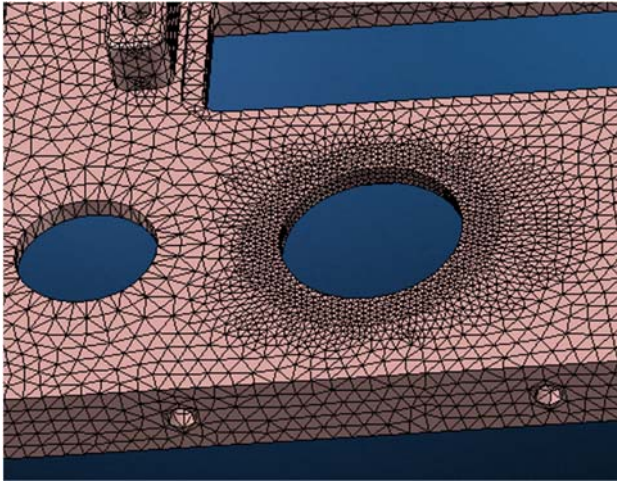


Figure 16: Washer face mesh around the left hole. Refined mesh around the right hole.

In areas where accurate stresses are needed, the elements should be of good quality. Many Solvers will happily run with poor quality elements, but under certain loads and with coarse meshes the element quality can skew the results. An example is the narrow plate shown in Figure 17.



Figure 17: sample model for influence of the mesh quality on the result

With rectangular elements, the result is correct (matches the analytically calculated solution). However if skewed quads or triangular elements are used, the resulting deformation deviates significantly from the correct result.

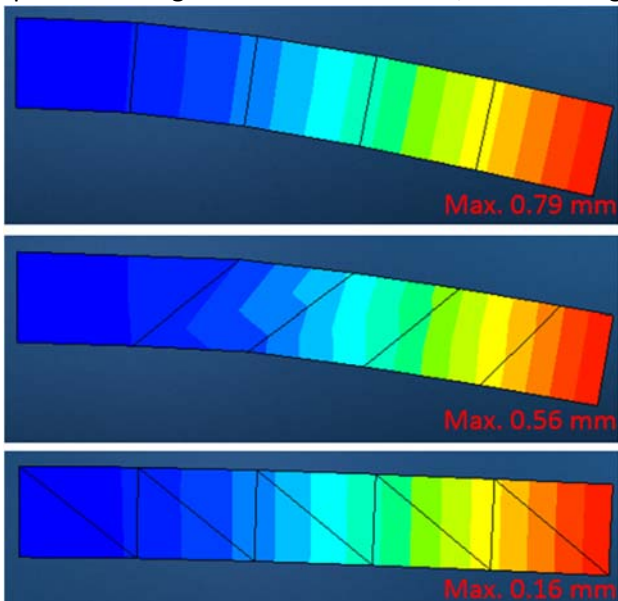


Figure 18: Results for the example model from Figure 17 (deformation exaggerated)

You can solve this problem in several ways:

- Use a finer mesh
- Ideally use square elements (high element quality)

The fig. 17 mesh is clearly too coarse, but serves to illustrate the point. In theory, you should use about 5 elements through the depth. But since such effects in real models may occur, you should pay attention to the quality of the elements. Figure 19 shows typical quality criteria.



Figure 19: The most common quality deviations in square elements

MSC Apex displays the element quality via color coding and uses four categories: invalid, bad, poor, and good. Various criteria can be displayed individually, or a general quality index, which is a weighted average of the individual criteria. MSC Apex can automatically improve the entire model, or you can move nodes manually, or split surfaces.

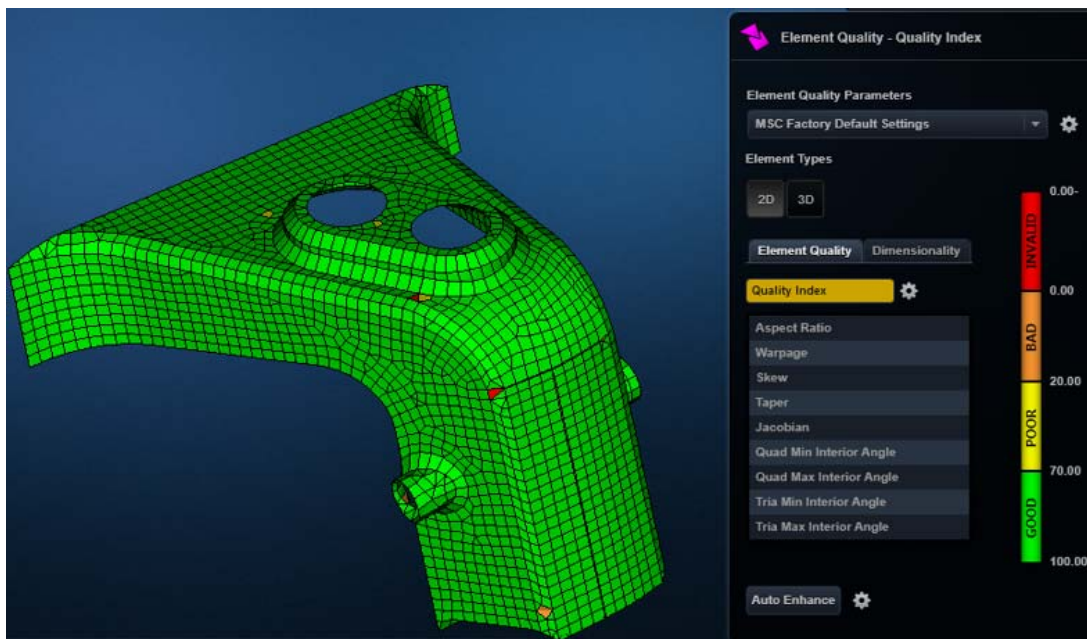


Figure 20: MSC Apex element quality index display

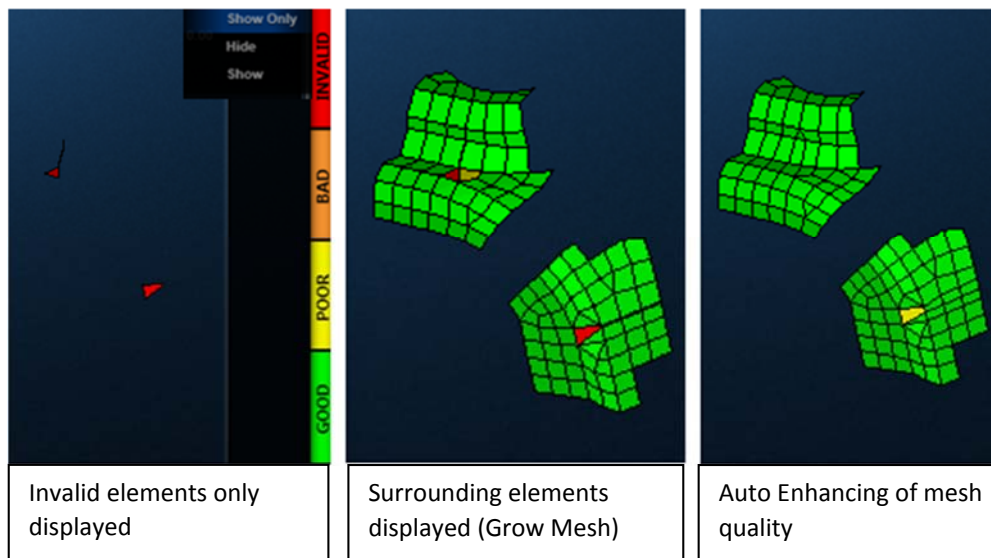


Figure 21: Improving the quality of the elements in MSC apex

Once we have created the mesh, and can make the remaining entries. In addition to a mesh, the following minimum input is required for a linear static FEA calculation:

- Material values
- Loads
- Restraints

MSC Apex indicates whether the model is ready for solving using red/yellow/green icons in the Analysis Readiness panel. Green means that all the necessary inputs are done, and the model is solver-ready. A yellow icon indicates that the model should be checked – for example, there may be poor quality elements. If red, the model cannot be run due to invalid element quality, missing constraints, missing material data, etc.

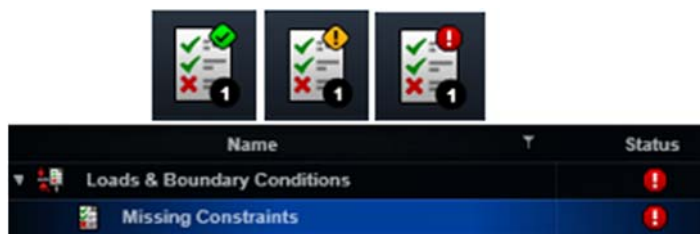


Figure 22: MSC Apex Analysis Readiness Indicator

Now the model may be solved and the results are displayed. There are various plausibility tests recommended, for example:

- Check the approximate magnitude of the deformation with a manual calculation
- Check that the applied load and reaction force vectors are sensible

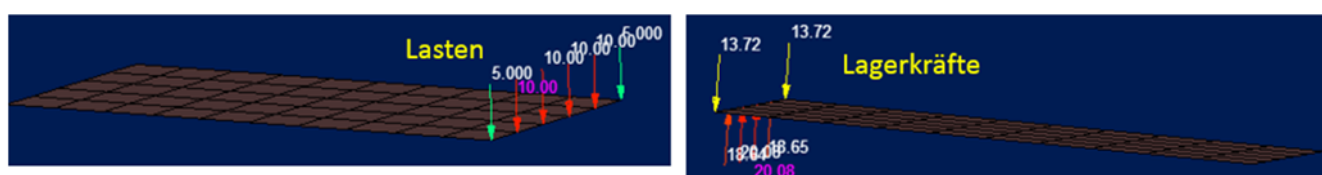


Figure 23: Comparison of the applied loads and the reaction forces

Stress results from any FE solver should always be examined with care. The solver calculates stresses for each individual element. Once these are displayed as a fringe plot, averaging at shared nodes between neighboring

elements takes place. This may mask significant jumps in stress from one element to the next. These stress discontinuities may be viewed in MSC Apex by either switching off averaging, or displaying stress differences. The model in Figure 24 shows the stress difference between the largest and smallest value calculated at each node of adjoining elements. If these differences are large you should refine the mesh. A difference value of 5-10% of the averaged stress value is a typical guideline.

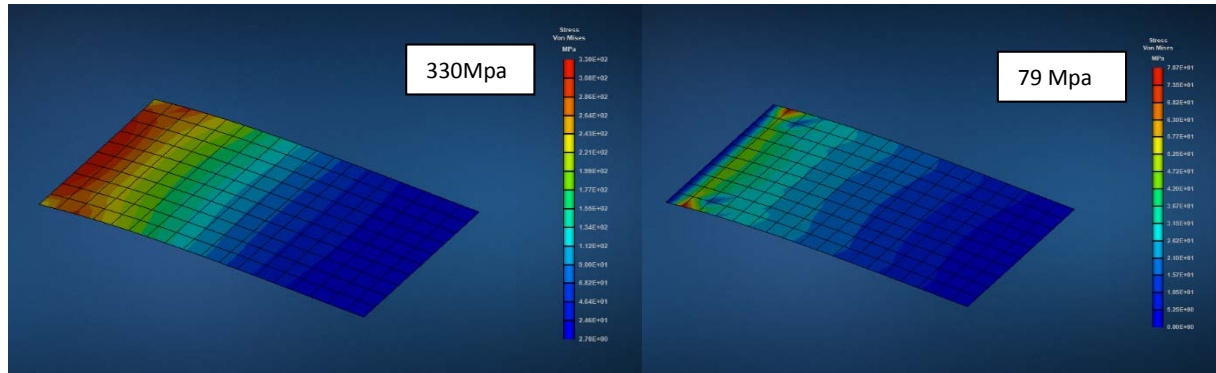


Figure 24: Stresses and Stress Differences

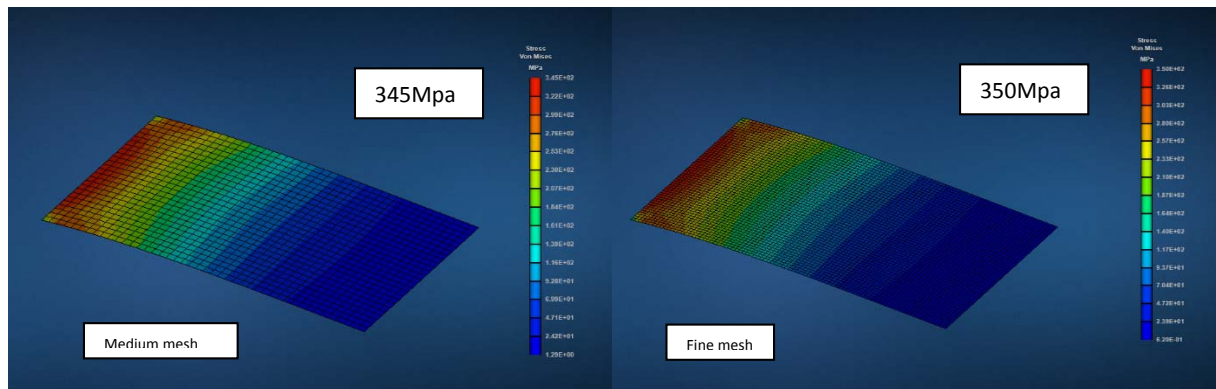


Figure 25: Stresses with highly refined meshes

For new models it may be worth testing how much the stress values change if one refines the mesh. Figure 25 shows the stress results for the model from Figure 24 with mesh refinement. The stresses converge to a value of 350 MPa.

For some models the stresses don't converge to a maximum value, but keep rising with higher mesh refinement. Causes include:

- Internal sharp edges (undercut)
- Load applied only at a single node
- Point contact

Figure 27 below shows an example in which the stresses around a hole did not converge.

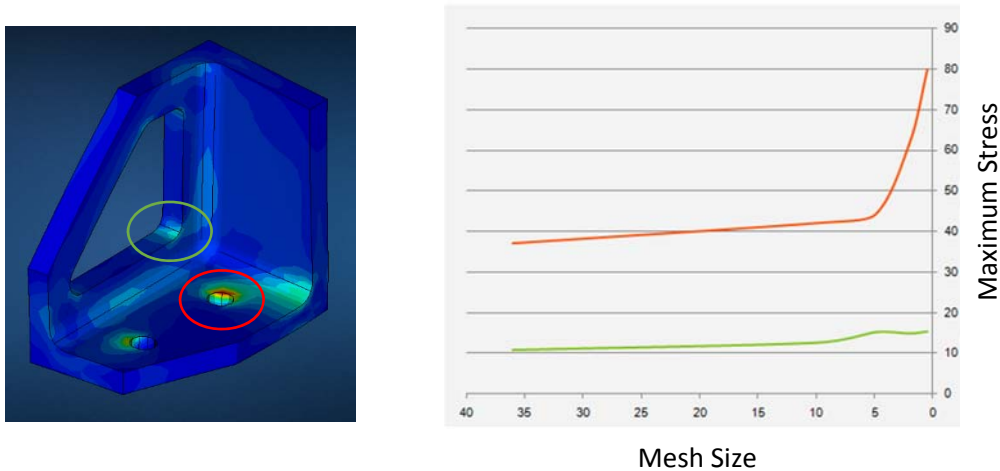


Fig. 27: Example of stress singularities

What can you do in such cases?

- Instead of using stress results, consider force results and draw conclusions about the durability
- Consider plastic material behaviour
- Despite the singularity, use the stresses for qualitative design comparisons (maintain a constant mesh size)
- Rate the stress according to FKM or similar methods, in which the stress at a defined distance from the edge is evaluated

For more information about creating FEM models <http://www.mscsoftware.com>

Find more information about MSC Apex <http://www.mscape.com>

The MSC Software corporate logo, MSC, and the names of the MSC Software products and services referenced herein are trademarks or registered trademarks of the MSC Software Corporation in the United States and/or other countries. All other trademarks belong to their respective owners. © 2016 MSC Software Corporation. All rights reserved.